



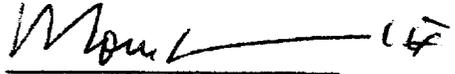
531337

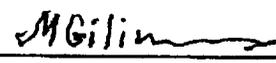
**HAMPTON UNIVERSITY**  
**NASA GRANT NAG-3-2249**

**ADVANCED METHODS FOR AIRCRAFT ENGINE THRUST  
AND NOISE BENEFITS. NOZZLE-INLET FLOW ANALYSIS**

**ANNUAL REPORT-  
SUMMARY OF RESEARCH**

**NASA Grant, NAG-3-2249**  
**3Year Granted Period**  
**Effective Date: 01/01/99**  
**Expiration Date: 12/31/01**

  
**Dr. Morris H. Morgan**  
**Principal Investigator**  
**Professor, Dean**  
**School of Engineering & Technology**

  
**Dr. Mikhail M. Gilinsky**  
**Primary Investigator**  
**Research Professor**  
**School of Engineering & Technology**

## **ADVANCED METHODS FOR AIRCRAFT ENGINE THRUST AND NOISE BENEFITS. NOZZLE-INLET FLOW ANALYSIS**

Three connected sub-projects were conducted under reported project. Partially, these sub-projects are directed to solution the problems conducted by the HU/FM&AL under two another NASA grants [1,2]. The fundamental idea uniting these projects is to use untraditional 3D corrugated nozzle designs and additional methods for exhaust jet noise reduction without essential thrust lost and even with thrust augmentation. Such additional approaches are: **a)** to add some solid, fluid or gas mass at discrete locations to the main supersonic gas stream to minimize the negative influence of strong shock waves forming in propulsion systems; this mass addition may be accompanied by heat addition to the main stream as a result of the fuel combustion or by cooling of this stream as a result of the liquid mass evaporation and boiling; **b)** to use porous or permeable nozzles and additional shells at the nozzle exit for preliminary cooling of exhaust hot jet and pressure compensation for non-design conditions (so-called continuous ejector with small mass flow rate; **c)** to propose and analyze new effective methods fuel injection into flow stream in air-breathing engines. Note that all these problems were formulated based upon detailed descriptions of the main experimental facts observed at NASA GRC. These projects are also based on several meetings and presentations with the NASA scientists, Dr. I.M. Blankson and R.C. Hendricks (GRC), C. S. McClinton (LaRC), US NAVY representative, C. Shnyder, and others. Basically, the HU/FM&AL Team has been involved in joint research with the purpose of theoretical explanation of experimental facts and creation of the accurate numerical simulation technique and prediction theory for solution current problems in propulsion systems solving by the NASA and NAVY agencies. This work is also supported by joint research between the NASA GRC and the Institute of Mechanics at Moscow State University (IM/MSU) in Russia under the CRDF grant [3]. The research is focused on a wide regime of problems in the propulsion field as well as in experimental testing and theoretical and numerical simulation analyses for advanced aircraft and rocket engines.

The FM&AL Team uses analytical methods, numerical simulations and possible experimental tests at the Hampton University campus. Below we will present some management activity and theoretical, numerical simulation results obtained by the FM&AL Team in the reporting period in accordance with the schedule of the work.

### **1) Sub-project A: Nozzle-Jet Mean Flow Numerical Simulation. One-Phase Flows**

In this sub-project several CFD codes for numerical simulation have been employed, improved and used for nozzle-jet mean flows numerical simulations. These codes are based on Navier-Stokes and Euler equations for viscous and unviscid flow simulations with and without non-equilibrium chemical reaction and relaxation processes. These codes are: two NASA CFL3D, VULCAN codes with different algebraic and differential turbulence models, ICASE FLUENT CFD code with LES option for turbulence modeling, Russian IM/MSU GROM code, and HU/FM&AL (KGG) codes. These simulations use different turbulence models, with and without taking into account non-equilibrium chemical reactions and relaxation processes.

The main achievements in this sub-project are:

a) New convergent and convergent-divergent triangular-round nozzle (T-R nozzle) was designed in accordance with the request of the NASA LaRC Hyper-X Program manager, C.S. McClinton. Experimental tests of such design will be conducted at the NASA LaRC and NASA Marshall Flight Center. Several modification of the design were proposed and drafted by the HU/FM&AL Team. In the first set of the T-R designs, smooth curves for the nozzle wall internal contour along the nozzle axis were used. These curves were composed by two coupled parabolae and its join two corresponding points: at the initial circle of the round cross section of the nozzle inlet (A) with the ending triangular cross section of the nozzle exit (B). In the second set of the T-R designs, each cross section along the nozzle X-axis is described by the supper-elliptical equation:  $[Z/A(X)]^{**N(X)}+[Y/B(X)]^{**N(X)}=1$  (\*) with the power N and the main axis A and B depending of the axial variable X so that, again, each circle's point at the nozzle inlet smoothly is joined with the point of triangular exit nozzle contour. For each nozzle design corresponding single and multi-grid meshes were constructed using NASA GREDGEN (Version 13) and own FM&AL codes for grids generation. Basically, the NASA CFL3d code was used for numerical simulation of flows inside convergent traditional axi-symmetric round and proposed triangular-round nozzles. The main goal of these simulations is to obtain correct numerical results for both nozzle designs, to choose the appropriate grid for the nozzle flow with the following stage of the problem solution: to determine and compare jet penetration and mixing efficiency for jets exhausted from round and T-R nozzles into supersonic cross flow. Appropriate grids for this problem were also constructed using the NASA GRIDGEN code. At present time, numerical simulation tests are conducting using CFL3d code running on the Hampton University 8th processors (ORIGIN) and NASA LaRC GEOLAB SGI UNIX based computer systems. Unfortunately, these simulations require essential CPU time for steady state designation for each case. For example, one case simulation for the T-R nozzle-jet flow exhausted to the rectangular combustion chamber requires several twenty-four hours run of the CFL3d code on these computers. The requested attempts to get access to the NASA Aimes supercomputers still did not have success through the Hyper-X Program officials because there are not enough its resources. We request for your support and help in this problem solution in the nearest future through current Program resources.

We are planning to test several other nozzle designs to examine efficiency of exhausted jet mixing and jet penetration into a supersonic cross flow, in particular, diamond shaped nozzle, Bluebell nozzle and others. The main researchers involved in this problem solution are a PI, Dr. Gilinsky (HU), Valery G. Gromov (IM/MSU), and Chuck S. McClinton and Carlos Rodrigez (NASA LaRC).

b) Analysis of stationary detonation and combustion at the internal designs of the axisymmetric or 2D Telescope nozzles with several internal designs ( from 1 to 4) was conducted in inviscid approximation. Numerical simulation results were obtained using 1st order Krayko-Godunov marching scheme. The Newman-During-Zeldovich (NDZ) combustion model with one exothermic chemical reaction was used (see [4] for details). Inside divergent portion of the supersonic conical round nozzle heat addition is realized at the internal designs behind oblique shock waves. As a result essential thrust augmentation is obtained. In the considered cases such thrust augmentation was obtained up to ~50% for axi-symmetric nozzle by comparison with the case without internal designs. Based on these results, invention disclosure application to NASA was prepared for submission. Also these results were a basis of the report submitted for the presentation at the 10th Hypersonic Propulsion Conference in Kyoto, Japan on April 27-29, 2001, and this report was accepted. At the present time we are examining these results with more accurate chemical reaction system for premixed hydrogen-air and hydrogen-oxygen mixtures using

the parabolized and full NSE approximations and Russian GROM and NASA VULCAN numerical codes. We have been invited two Russian scientists from Institute of Mechanics at Moscow State University, Dr. Valery G. Gromov and Dr. Vladimir I. Sakharov. These scientists are conducting research at Hampton University FM&AL during January 20- March 30, 2001. Necessary documents were sent to US Embassy in Moscow and to Drs. Gromov and Sakharov. Note, that their trip to Hampton University is simultaneously supported by the CRDF grant RE1-2068 [3]. Some results of this joint research will be presented at the 37<sup>th</sup> AIAA/ASME/SAE/ASEE Joint Propulsion Conference in Salt Lake, UT, USA on 8-11 July, 2001.

## **2) Sub-project B: Nozzle-Jet Mean Flow Numerical Simulation. Two-Phase Flows**

Two main problems were proposed to analyzed in this sub-project: a) To create numerical codes for two-phase flow simulations using continuous representation for each phase (two-liquid approximation), and discrete-trajectory approximation with continuous carrying gas phase and discrete representation of solid or liquid particles moving along their trajectories. These two approaches were supposed to be modernization of existent ICASE/Fluent and MSU/GROM numerical codes (full NSE and parabolized versions) transforming them to two-phase (and multi-phase) versions for both approximate models. b) To conduct numerical simulation of gas jet injection carrying solid or liquid particles. The simulations include many non-equilibrium relaxation processes for particles and chemical reactions in the carrying gas. Physico-mathematical models of these processes are using from the previous research conducted by a PI and presented, in particular, in [4,5].

The main achievements in this sub-project are:

In this sub-project, preliminary work was conducted. The numerical code based on the 1-st order Godunov scheme for an inviscid unsteady gas flow numerical simulation was improved for discrete-trajectory model inter-phase interaction by adding several subroutines for particle parameters and trajectories calculation with corresponding changes in the conservation laws for carrying gas. This code was used for numerical simulation of two-phase jet injection against supersonic gas flow around butt-end. The main differences with previous numerical simulations conducted in Russia [4] were an application more accurate resolution by application of fine grids and application more effective smoothness limiters. However, with increase of grid-points number some oscillations were observed in gas parameters distribution. In present time, we are working to eliminate such instability. Also we are planning to apply similar approach for marching Krayko-Godunov scheme. Currently, Drs. Gilinsky, Gromov and Sakharov and HU Graduate and undergraduate students, K. Patel, C. Coston are involved in these problems solution. During visit Russian scientists, will also work with this problem.

## **3) Acoustics Calculation. Sub-project C: Semi-Direct Numerical Simulation Method for CAA**

To get realistic distribution of acoustic sources in a turbulent flow, the LES approach is employed. This approach represents a reasonable compromise between the direct numerical simulation (DNS) method (unavailable for high Reynolds numbers) and the unsteady Reynolds average numerical simulation (RANS) method (where the turbulence and acoustic time may be mixed up). To avoid extraction of acoustic sources from the turbulent fields, we use the same computer code for CFD and acoustic simulation. To achieve this goal, we expand our higher-order compact aeroacoustic parallel code to

compute background mean flows. Then, we suppose to compare this approach with the use of the second-order FLUENT CFD code with the large eddy simulation (LES) option for turbulence modeling, extraction of acoustic sources from the computed turbulent field and use of the high-order code for aeroacoustics only.

The main achievements in this sub-project are:

For the Bluebell nozzle design unstructured grid was constructed using FLUENT grid generation option for given surface data. The external nozzle surface was chosen so that Bluebell nozzle exit has sharp, infinitesimal thin edge. The computational domain include internal region inside a nozzle and jet region with the surrounding space limited by 15 calibres downstream, 6 calibres in radial direction and 3 calibres upstream. The first numerical simulation tests were conducted. The main researcher responsible for this sub-project fulfillment is Dr. Alex Povitsky with participation of Dr. Jay Hardin as a consultant.

#### 4) Experimental Tests at the HU Low Speed Wind Tunnel (LSWT)

**Sub-Project D: Mechanical Jet-Mixer Noise Suppressor.** All jet noise suppression methods are connected with an influence on the mixing layer. Mixing enhancement leads to reduction of almost all jet noise components which are generated by the small scale turbulence as well as the large scale turbulence. Therefore, such enhancement can be achieved by creating artificial additional mixing sources in the initial part of the mixing layer, i.e. directly at the nozzle exit edge. We proposed to apply as such sources screws of small size relative to the nozzle exit radius. There are many variations in selection of the screw orientations, geometric parameters and shapes that define the kind of vortices which we would like to generate in the mixing layer, counter-rotating or streamwise (longitudinal) vortices.

In the proposed devices, the screw can also be rotated using a small motor for each screw or one for all. For example, they might be installed inside the space between the internal and external surfaces of the engine nozzle. In the laboratory experimental test, we will mount these motors outside the nozzle upstream of the nozzle exit such that the presence of these devices will not influence the mixing process in the mixing layer. For small-scale experiments in the HU LSWT model, we will use motors that have been developed for motor boat toys or similar. Note that, in both cases, for cross and streamwise mixers, our Möbius shaped screw can be applied and tested as an additional examination of its efficiency.

The main achievements in this sub-project are:

This sub-project is closely connected with two other projects conducted under the NASA grants [1,2]. Under these grants teaching of several courses and training in the field of fluid mechanics and acoustics must be provided at Hampton University. Therefore, at present time, HU wind tunnel measurement system is improving. New pressure system will install in the experimental hall. It will allow simultaneous static pressure measurement in 24 points using transducer system and automatic data processing using FM&AL computers. Also, this system is necessary for efficient experimental tests of jet-mixer noise suppressor. All elements of this system are ordered, and we expect to get its soon and to start installation. Two convergent nozzles with transition portion are making from still and wood by the

graduate students. Its will be mounted into the HU LSWT for research and training, in particular, for the sub-project F fulfillment.

**Sub-Project E. Piezoelectric Transducers Application.** Piezoelectric and magnetostrictive transducers help in detecting and generating acoustic waves over a wide bandwidth. The turbulence of exhaust gases generates a wide band of noises in the nozzle area. The low frequency components of the noise travel over a long distances and pose severe noise pollution problems.

The structure of nozzle has its own sets of resonance frequencies when it vibrates in different modes. The forceful movement of flue gasses out of the nozzle induces such resonant vibrations along the edges of nozzle and aggravates the situation of noise pollution.

The proposed measure of noise reduction using acoustic transducers around the nozzle involves positioning a set of acoustic transducers around the nozzle, which will sense the vibrations around various segments of the nozzle and apply forced vibration with equal magnitude at opposite phase to reduce the acoustic noise extensively. In this way, it is expected that low frequency noise ( $< 100$  Hz) can greatly be reduced. Low frequency noise propagates longer distances and induces vibration on the walls and windows of the buildings. However, the annihilation of low frequency signals require the generation of high power and will pose a practical limitation on the power handling capacity of the transducers mounted around the nozzle. The transducers have a limitation on the maximum temperature that they can withstand. Therefore, the mounting of transducers will involve the necessary stand-off and acoustic couplings to ensure that the transducers stay well within the operating range of temperatures.

The main achievements in this sub-project are:

All necessary elements of the piezoelectric transducers system were determined as well as the prices list and companies distributed its. Corresponding purchasing orders were submitted to HU administration. The total amount of this system installation is  $\sim 4,5K$ , and we are waiting the 3<sup>rd</sup> year financial support for the current grant to provide this system installation. The nozzle fabrication described above is for testing of this piezoelectric transducers system efficiency. The main researcher, responsible for this sub-project fulfillment, is Dr. V. Jagasivamani and several undergraduate students from Electrical Engineering and Aviation Departments are involved in this research sub-project. The review and the status of this work will be prepared on April, 2001.

### **SOME ADDITIONAL INFORMATION ABOUT RESEARCH ACTIVITY**

(FROM THE US-FSU TEAM QUARTARLY REPORT TO THE CRDF)

Additional research conducted jointly with Russian team from Institute of Mechanics at Moscow State University under CRDF [3] and current grant. Basically, the main results were obtained by Russian team with supervision and control of US team (Drs. I.M. Blankson, R.C. Hendriks and M. Gilinsky).

In the reported period the research was conducted in two directions:

- 1) 3D calculations taking into account turbulence and experimental researches of drag reduction effect for blunt bodies with needles;
- 2) Numerical investigations of flow in Half-Duct Combustor System with use hydrogen as injected fuel.

The review of researches on the flows past the blunt bodies with a needle in a nose (with a forward fly-off zone) has been prepared. The publications in the leading scientific log-books of USA, Russia and other countries, data received in a network Internet, for period from the end of fifties till now are used. The review contains results of the theoretical and experimental researches and includes about 60 activities from USA and 20 – from Russia. Table of contents involves sections: the schemes of flows, aerodynamic characteristics (drag, lift, center of pressure position), stability, heat fluxes, non-stationary regimes. Influence of the shapes of the blunt bodies and needles, Mach and Reynolds numbers and practical applications of the similar shapes are considered.

Pursuant to the plan of experimental activities:

- the designing of models has been carried out;
- the models of cylindrical bodies with flat forebody (model diameter is 80 mm) and needles ( $n = 1, 5, 8$ ; needle elongation  $L = 0.5, 1.0, 1.5$ ) have been made;
- calibration of 3-component scales, the measuring equipment have been prepared.

The testing experiments of flows past cone and cylindrical body with flat forebody have been carried out. The methodical tests for blunt-shaped cylinder with needles ( $n = 0, 1$ ;  $L = 1.0, 1.5$ ) have been carried out at number Mach,  $M=3.0$ , to obtain aerodynamic characteristics and flow photos. The data of the present experiments correspond to the published results.

2D and 3D numerical simulations of supersonic flow over wall-end with single needle ( $M=3$ ,  $L=0-1.5$ ) have been performed taking into account  $k-\omega$  turbulence model in the framework of the Navier-Stokes equations. Influence of turbulence on the solutions has been investigated. Using the turbulence model during the calculations changes solutions and leads to the existence of the steady-state regimes for some parameter ranges.

The test of new Fortran codes for Navier-Stokes modeling of gas-dynamic and chemical processes in Half-Duct Combustor System has been performed. The testing calculations of flows in plane channel with various rectangular uniform grids, difference scheme parameters, channel geometry and input data have been done. It has been analyzed how the numerical parameters affect accuracy of numerical results to select their optimal values. The effective generalized space-marching procedure previously used for external flow calculations has been developed to obtain internal flowfields in a channel. Test results have shown the high performance of this approach for considered type flow computations. The codes for visualization of numerical results have been created. The parametric calculations of flow in air-hydrogen combustor for various input conditions and channel geometry were started on the base of developed numerical technique to optimize the fuel injection location.

In next quarter (01/01/01-03/30/01) it is supposed:

- to manufacture models to measure pressure distributions on the blunt bodies with the spikes;
- to manufacture models to provide weight tests in the experiments with an expiring jet of fluid or firm particles from blunt bodies;

- to perform the experimental investigations (weight tests, flow visualizations) of flows past blunt bodies with spikes having flat face and hemispherical bluntness. The 1, 5, 9 spikes of various length ( $L = 0, 0.5, 1.0, 1.5 D$ ) will be placed on the frontal body surface (model diameter is 80 mm; spike diameters are 0.07, 0.1, 0.13 D). The tests will be carried out at Mach numbers  $M = 0.6 - 1.2, 2.0, 3.0$ .
- to perform parametric calculations of flow over bodies with needles taking into account  $k-\omega$  turbulence model;
- to continue jointly with US co-investigators parametric calculations of flow in air-hydrogen combustor for various input conditions to optimize the fuel injection location
- to transfer the developed codes and computation technique to US co-investigators.

### **THE MAIN OUTCOMES DURING REPORTING PERIOD**

1. The Primary Investigator, Dr. Gilinsky was awarded by the US patent [6] on July 19, 2000.
2. The HU/FM&AL was awarded by the NASA grant NAG-3-2495 [2] on October, 2000.
3. The paper presentation at the 36<sup>th</sup> AIAA/ASME/SAE/ASEE Joint Propulsion Conference, 17-19 June, 2000, Huntsville, Al by Gilinsky, M., and Blankson, I.M., 2000, entitled: Internal Designs Application for Inlet and Nozzle Aeroperformance Improvement, AIAA Paper #00-3170 [7].
4. Submission of the abstract for the presentation at the 37th AIAA/ASME/SAE/ASEE Joint Propulsion Conference, in Salt Lake City, UT, being held 08-11, July, 2001. The abstract was entitled: "Shock Waves Mitigation at Blunt Bodies Using Needles and Jets against a Supersonic Flow" by M. Gilinsky I.M. (HU), Blankson, V.I. (GRC), Sakharov and A.I. Shvets (IM/MSU) (in pending).
5. Submission of the abstract for the presentation at the 10<sup>th</sup> International Space Planes and Hypersonic System and Technology Conference, in Kyoto, Japan, being held 27-29 April, 2001. The abstract is entitled: "Corrugated and Composite Nozzle-Inlets for Thrust and Noise Benefits" by M. Gilinsky (HU) and I.M. Blankson (GRC) (accepted).
6. Preparation for publication as a NASA Report of the review entitled: "Spiked Blunt Bodies" by M. Gilinsky, A.I. Shvets, and I.M. Blankson.
7. New course for HU students of the School of Engineering and Technology entitled: "Advanced Aerodynamics and Aircraft Performance" lectured on Spring semester by the FM&AL researchers Dr. M. Gilinsky and N. Sckholnikov. Training in experimental tests using HU LSWT.
8. Installation in the FM&AL computer system: second software TECPLOT 8.0 for UNIX SGI and PC Dell computers, and 2D GRIDGEN for the UNIX SGI as well as installation of two free NASA codes 3D MAG and VULCAN.
9. Numerical simulation results using NASA, FM&AL and IM/MSU codes based on Euler and Navier-Stokes equations, for one-phase gas flows.

## REFERENCES

1. Numerical Simulation of One- and Two-Phase Flows in Propulsion Systems, 2000-02, MURED Program, 3 year FAR Award, NAG-3-2422, PI- M. Gilinsky.
  2. Corrugated and Composed Nozzles Application via UEET Program, 2000-2002, MURED Program, 3 year Award, NASA grant NAG-3-2495, PI-M. Gilinsky.
- Shock Waves Mitigation at Blunt Bodies using Needles, One- and Two-Phase Jets against a Supersonic Flow, 2000-01, CRDF grant RE1-2068, Co-PI: Blankson I.M., and Shvets, A.I.
4. Gilinsky, M., 1992, Advanced Gas Dynamics for Aerospace and Defense Applications, Moscow State University, Institute of Mechanics R&D, Global Consultants Inc., Alexandria, VA, 249 p.
  5. Gilinsky, M, Seiner, J.M., et al., Supersonic Gas-Dispersion Jets: Models and Applications, 1994, AIAA Paper 94-0135, 32nd Aerospace Sciences Meeting and Exhibit, Jan. 10-13, 1994/Reno, NV, 24p.
  6. Seiner, J.M., and Gilinsky, M., 2000, Undulated Nozzle for Enhanced Exit Area Mixing, US Patent #6,082,635.
  7. Gilinsky, M., and Blankson, I.M., 2000, Internal Designs Application for Inlet and Nozzle Aeroperformance Improvement, AIAA Paper #00-3170, 36<sup>th</sup> AIAA/ASME/SAE/ASEE Joint Propulsion Conference, 17-19 June, 2000, Huntsville, Al.